

Design, Development and Experimental Evaluation of Reverse Flow Combustor

Muralidhara H S, Muthuselvan G, Vinod Kumar Vyas,
Jeyaseelan A R, Rajeshwari N, Hariharan and Satish Kumar M
Propulsion Division, National Aerospace Laboratories, Bangalore-560 017
murali@nal.res.in

Abstract

A Typical reverse flow combustor is developed for an application in a small gas turbine engine. This is designed for 4 kg/s air flow rate, 6 bar pressure and 1000K exit temperature. This is designed from first principles and fine tuned with Computational Fluid Dynamics (CFD) technique. Combustor is developed and experimentally evaluated in a test rig of Combustion Laboratory of National Aerospace Laboratories (NAL). Inlet to the test rig is compressed dry air at ambient temperature. Combustor performance parameters such as combustion efficiency, total pressure loss and exit temperature pattern factor is presented. Effects of equivalence ratios on these combustion performance parameters are discussed.

Keywords: CFD, Combustion efficiency, Total pressure loss, Circumferential pattern factor, Reverse Flow Combustor(RFC)

1. Introduction

There are many types of combustor configuration available for the use in small gas turbine turbo machinery. But the reverse flow combustor(RFC) offers many advantages[1,2] when compared to other types. Reverse-flow combustor can reduce the length of the gas turbine. Reduced length will allow single shaft to sit on two bearings instead of three. This will reduce the vibration and maintenance problems. The reverse-flow layout effectively uses the air flow to cool down the combustor liner. The absorbed heat by the air is returned back to the system. In other words, the reverse-flow combustor cooling process is actually a preheating process of the air. The reverse-flow process also allow warmer air to serve as the dilution air to control the NO_x formation instead of using other energy to preheat the dilution air or use cold air which could quench the flame and produce CO.

In the present work, reverse flow combustor is designed from the principles outlined by Lefebvre[1], Mellor[3] and Zuyev[4]. CFD is used as a design tool[5] for fine tuning the combustor configurations[6,7]

Considerable amount of numerical work has been done to understand the flow behaviour of reverse flow combustor. D.S. Crocker and C.E. Smith[7] performed numerical study to understand the effect of advanced dilution hole concept (injecting dilution air jets with high circumferential component) on pattern factor. From the numerical work, the authors found optimum dilution hole spacing, jet angles and mass flow split to improve the quality of temperature distribution at exit. Yufeng Cui et al [8] performed numerical investigation on stagnation point reverse flow combustor. The numerical results indicated that increase in fuel and air mixture injection velocities increases the CO emissions,

decreases flame temperatures and NO_x emissions. Also the authors observed that as the equivalence ratios increase from 0.5 to 1, the NO_x emissions always increase. However, the CO emission decreases first, reaching the minimum value at equivalence ratio of 0.58, and then increases. Satish Undapalli, Srikanth Srinivasan, Suresh Menon [9] performed numerical analysis on stagnation point reverse flow (SPRF) combustor. The flow features and the combustion characteristics of both premixed and non-premixed modes are studied using LES. It is shown that premixing with hot products is the process that enables stable operation at a very lean equivalence ratio.

Experimental studies with combustion on reverse flow combustors are limited to basic studies[10-11]. Therefore one of the main objectives of this study is to analyse the combustor performance of reverse flow combustor.

2. Design

The Reverse flow combustor is designed for 4 kg/s at 6 bar inlet pressure with a temperature rise of 700 deg C across the combustor. Inlet air is taken at ambient temperature. Jet A-1 is used as fuel. A simplex atomizer which has been developed for reverse flow combustor application in this laboratory [12-14] is used for fuel atomization. Only eight atomisers are used to keep the design simple. After designing from the first principles, combustor drawing is prepared and is shown in Figure.1. Six rows of cooling holes are provided in outer liner and three rows are provided at inner liner. Z-strips are provided at cooling hole rows such that cooling air is glides along the liner providing better cooling efficiency. At the turning area of combustion flow path, sufficient care is taken to protect the outer liner

material by providing adequate cooling rows of holes.

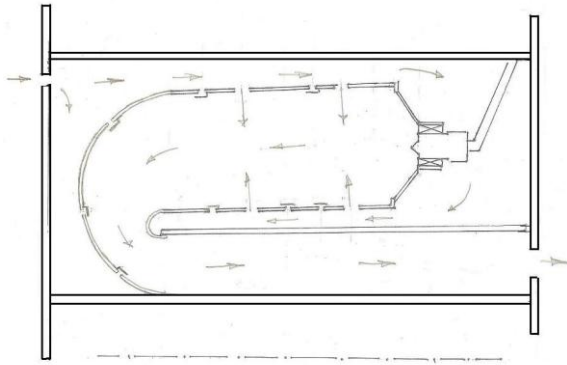


Figure.1 Diagram of reverse flow combustor

3. CFD

ANSYS Fluent Version 14 is used for CFD and utilised extensively to fine tune the combustor geometry. For this purpose only 45 degree sector is taken as there are only eight atomisers used. CAD package “Solid Works” is used to model the combustor and around 3.6 million tetrahedral grids were generated using ANSYS Meshing. $k-\epsilon$ model is used turbulence modelling and non premixed - equilibrium combustion model is used. For air and fuel “mass flow” inlet conditions used and exit “pressure outlet” conditions are used. Figure 2 (a) shows CAD model generated, Figure 2(b) shows sub assembly of liner, dome, swiler & atomiser and Figure 2(c) shows the tetrahedral grid generated.

CFD has been utilised to optimise the combustor geometric configurations. One of the major challenges in reverses flow combustor (RFC) is the alignment of primary and secondary jets. Both must hit each other in order to get satisfactory recirculation zone, mixing and good combustion. In RFC, flow in outer annular passage in one direction, where as in inner annular passage, flow is in reverse direction. When flow enters from annular passage to combustor zone, naturally, those two jet may not hit each other. This can be effectively optimised by CFD. By properly varying the hole positions, correct flow patterns can be obtained. This is demonstrated in Figure 3 for primary holes. Figure 3(a) shows flow pattern before fine tuning and Figure 3(b) shows flow pattern after optimising the combustor hole positions.

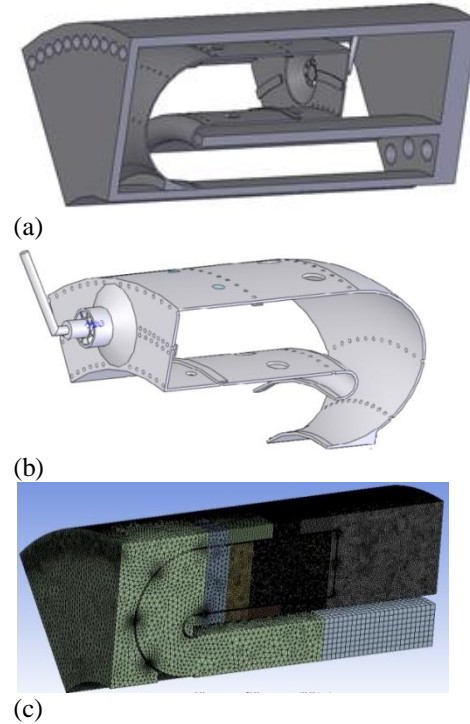


Figure.2 (a) Computational domain – CAD Model (b) Liner with atomizer and swirler assembly and (c) Grid generated

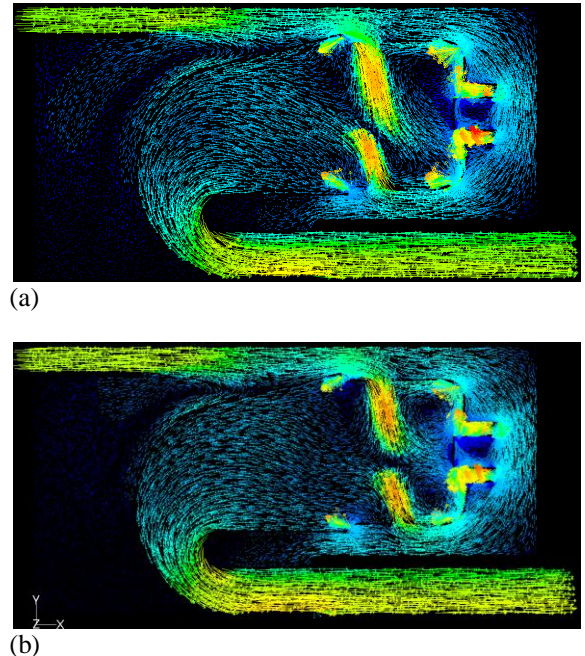


Figure.3 Velocity vectors for reverse flow combustor. (a) Primary jet not aligned (b) Primary jet aligned

In this combustor, several rows of holes are provided for combustor liner cooling. Various liner cooling configurations were considered for hot flow CFD analysis, finally one cooling hole configuration has been identified for effective liner cooling. Optimised combustor configuration is shown in figure 4. Figure 4a shows mid plane temperature contour of reverse flow combustor. it shows high temperature zone in the primary zone of combustor. Figure 4b shows mid plane velocity vector contour of reverse flow combustor. This shows the presence of central toroidal recirculation zone and, penetration of air through primary and dilution hole.

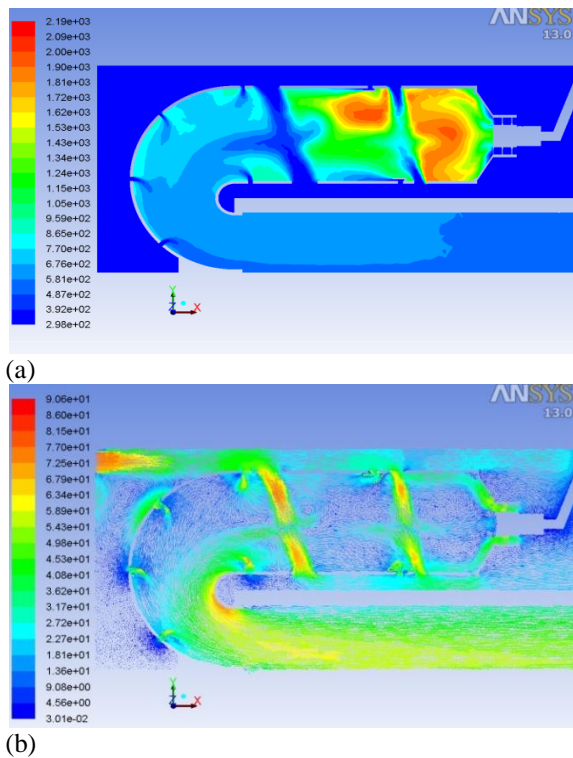


Figure.4 (a) Temperature contour at the atomiser plane for air to fuel ratio of 75. (b)Velocity vectors at the a atomiser plane for t air to fuel ratio of 75

4.1 Experiments

Figure 5 shows the layout diagram of experimental setup. RFC is mounted on 8 inch airline. Dry air at ambient temperature is supplied to the test rig. Gate valve is used to open & close the air supply and by globe valves is used to control the desired flow rate. Standard orifice plate is used with D & D/2 tapings for air flow measurements. Rake with five total pressure is used to measure inlet total pressure. Thermocouple is also mounted to measure inlet temperature. At the exit of RFC, 18 thermocouples and two pressure probes are mounted to measure exit temperature and pressure. Fuel is supplied to the RFC from an over head tank. Reciprocating high pressure pump is used to supply the fuel at any desired pressure and flow rate. This is controlled by

globe control valve. Excess fuel is sent back to the tank by a separate bypass line, which is also controlled by globe control valve. Figure 6(a) shows cross sectional view if RFC and Figure 6(b) shows liner assembled with atomiser, swirler and dome. Figure 7 shows the actual hardware mounted on test rig. This photograph was taken during experiment, visible flame at RFC exit can be observed.

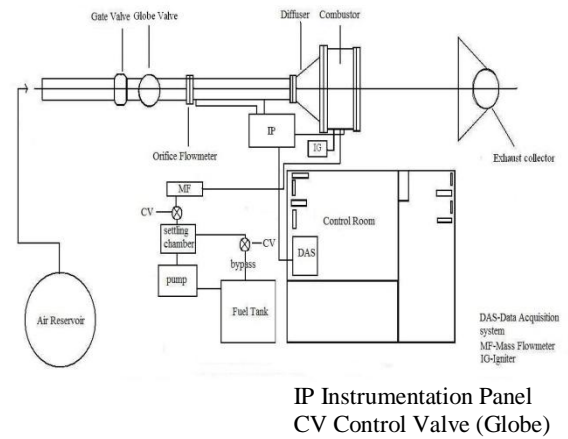


Figure.5 Experimental layout of RFC rig setup

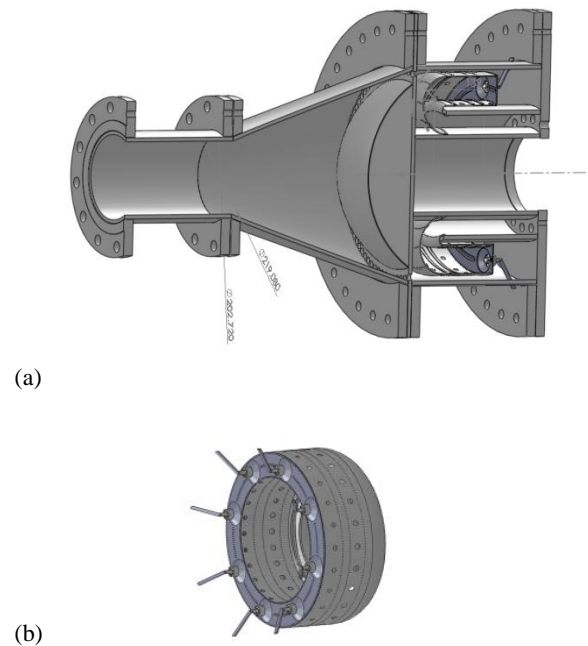


Figure.6 (a) Cut section of RFC (b) Liner with fuel injector



Figure.7 Experimental test rig for RFC

4.2 Instrumentation

An integrated measurement system is developed in LabVIEW using NI PXI system for testing the reverse flow combustor. This system has the capability to measure 25 temperatures, 12 pressures and fuel mass flow. Pressure transmitters with $\pm 0.5\%$ accuracy are used for pressure measurements. K Type thermocouples are used for all temperature measurements. Coriolis mass flow meter with $\pm 0.25\%$ is used to measure fuel flow rate. The software of the system acts as a brain which integrates all the measurement to display and store. The mass flow of the fuel and air is controlled by an independent unit which helps in injecting the fuel and air in to the combustion chamber. The combustor chamber is ignited using a standalone unit. The important parameters like air flow rate, equivalence ratio, average of the inlet pressure and combustor performance parameters are calculated using the measured parameters and stored.

4.3 Performance Parameters

Following parameters are used in this paper to evaluate the performance of the RFC

Equivalence ratio ϕ : This is defined as ratio of actual fuel-air ratio to stoichiometric fuel-air ratio. This is 0.0685 for the fuel considered.

Combustion efficiency η : It is defined as the ratio of heat released in combustion to the heat available in fuel. In this paper, following equation is used to calculate combustion efficiency

$$\eta = (T_{\text{exit avg}} - T_{\text{inlet}}) / (T_{\text{exit theoretical}} - T_{\text{inlet}}) \quad (1)$$

Here,

$T_{\text{exit ave}}$ is measured average exit temperature

T_{inlet} is measured inlet temperature and

$T_{\text{exit theoretical}}$ is theoretical temperature as obtained by NASA CEA Program [15]

Circumferential Pattern Factor (CPF): This is defined as

$$\text{CPF} = \frac{T_{\text{max}} - T_{\text{exit avg}}}{T_{\text{exit avg}} - T_{\text{inlet avg}}} \quad (2)$$

Here, T_{max} is the maximum measured temperature at combustor exit

Total Pressure Loss: This is defined as

$$\text{Tot. Pr. Loss} = \frac{(P_{0, \text{Inlet Average}} - P_{0, \text{Exit Average}})}{P_{0, \text{Inlet Average}}} \quad (3)$$

Here P_0 refers to measured total pressure

4.4 Results and Discussions

During the experiment, combustion was initiated with proper fuel-air ratio, later air flow rate was kept constant. Then fuel flow rate was decreased slowly till blowout condition.

Figure 8 shows the effect of equivalence ratio on combustor average exit temperature for different values of inlet air flow rates. As expected, it is observed that with increase of equivalence ratio, combustor exit average temperature increases. All the experiments have been conducted within equivalence ratio 0.25, otherwise temperature could increase beyond safety limits (1000°C).

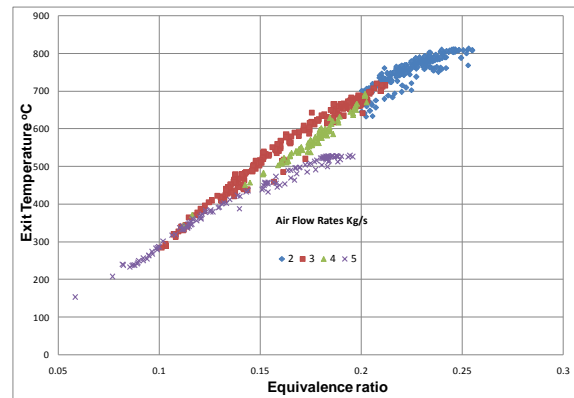


Figure.8 Effect of equivalence ratio on combustor exit temperature

Fuel atomization plays a vital role to improve the combustion efficiency. Figure 9 shows the effect of equivalence ratio on combustion efficiency. This shows that at lower flow rates (2 & 3 kg/s) combustion efficiencies are lower, due to poor mixing of air and fuel and poor atomization of fuel at lower pressure. At higher air flow rates (4 & 5 kg/s), better combustion efficiencies are achieved. Combustion efficiency could be further improved by using more number of atomisers.

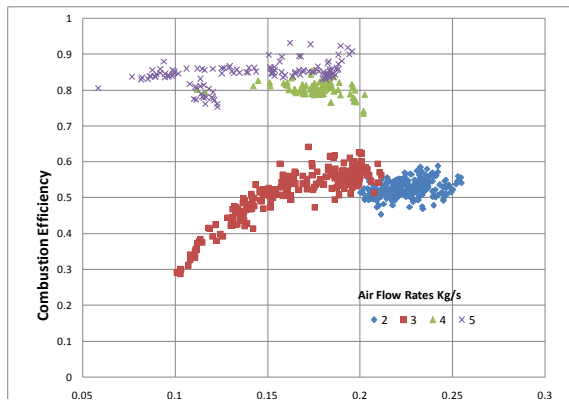


Figure.9 Effect of equivalence ratio on combustion efficiency

Figure 10 shows the effect of equivalence ratio on circumferential pattern factor (CPF). At lower air flow rates (2 & 3 kg/s), CPF is obtained in the range of 0.15 to 0.2 and at higher air flow rates (4 & 5 kg/s) it is in the range of 0.3 to 0.35. higher CPF at higher air flow rates is due to higher operating pressure. Generally for RFC, it is expected to get a better CPF (Less than 0.15). But in this design only 8 atomiser has been used, therefore CPF has increased.

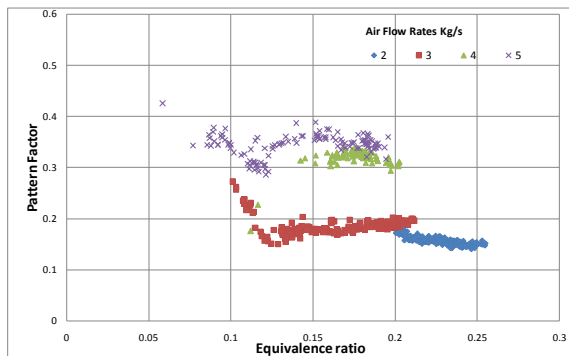


Figure.10 Effect of equivalence ratio on circumferential pattern factor (CPF)

Figure 11 shows the effect of equivalence ratio on percentage total pressure loss. It is noticed that, at higher the air flow rate, higher the total pressure loss is observed. This is expected as pressure loss is proportional to kinetic energy of the flow. It is also noticed that, for each flow rate, with increase of equivalence ratio, total pressure loss decreases. This is because, during experiment it has been observed that, even though air flow rate is expected to be constant, with increase of combustion intensity, there was slight reduction in air flow rate. Due to this reduction air flow rate, kinetic energy is reduced thus there is fall in total pressure loss.

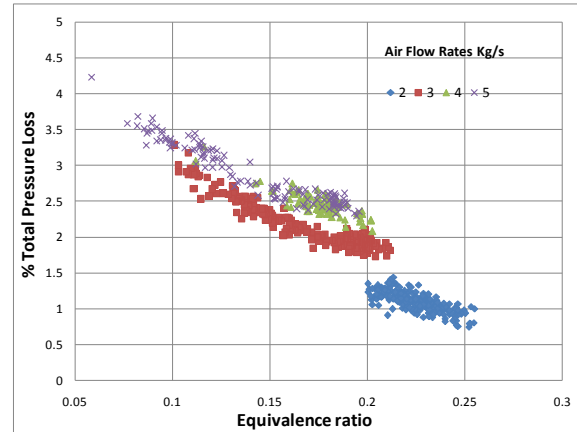


Figure.11 Effect of equivalence ratio on percentage total pressure loss

5. Conclusions

Typical RFC is designed and fine tuned with CFD techniques. RFC is fabricated and experimentally tested. Effect of equivalence ratio on exit temperature, combustion efficiency, circumferential pattern factor and total pressure loss is evaluated and following observations are made.

1. The exit temperature increases with equivalence ratio. For the given equivalence ratio with increase of airflow rate the exit temp slightly reduces.
2. Combustion efficiency increases with increase of equivalence initially and then flattens. For given equivalence ratio, with increase of air flow rate, combustion efficiency increases.
3. Circumferential Pattern Factor decreases slightly and then remains constant, with increase of equivalence ratio. For a given equivalence ratio with increase in the air flow rate the pattern factor increases.
4. For the given equivalence ratio, with increase of air flow rate, pressure loss factor slightly increases

Acknowledgement

Authors acknowledge Dinesh Kanta (NALTech Pvt. Ltd.) for his CAD support. Authors also thankful to Sudeep.P, Shanmukhi Krishna A, Rohit Kumar M and Venkatesh Y, BE Project students from Vardhaman College of Engineering, Hyderabad for their assistance during experiment and data analysis.

References

- [1] Books: Lefebvre A H, *Gas Turbine Combustion*, 1983 (Hemisphere, Washington).
- [2] Arunachalam P *Selection of a suitable combustion system for a small gas turbine engine*, defense science journal, Vol 38, No.4, 1988, pp381-396
- [3] Books: Mellor A M, *Design of Modern Turbine Combustors*, 1990, Academic press, London
- [4] Books: Zuuyev V S, and Skubachevskii L S, *Combustion Chambers for jet Propulsion engines*, 1964, Pergamon Press
- [5] Muralidhara.H.S, B.B.Chinmaya Kumar and T.R.Shembharkar, "*CFD Analysis of Gas Turbine Combustor: A CAD To Mesh To CFD Approach*", "6th National Conference on Air Breathing Engines and Aerospace Propulsion", (NCABE), 10-11, Jan. 2003, Bangalore.
- [6] Muthuselvan G, Muralidhara H S, DineshKanth T P, LakshmiNarayana Y "*Computational Study of Reverse flow combustor for small Gas turbine application*", National Propulsion Conference Chennai Paper No- 14008
- [7] Crocker D S, and Smith C E, "*Numerical Investigation of Enhanced Dilution Zone Mixing in a Reverse Flow Gas Turbine Combustor*", J. of Engg. For Gas Turbine and Power, April 1995, Vol 117 pp 272-281
- [8] Yufeng Cui, Xuan Lu, Gang Xu, Jianli Chen, ChaoqunNie, and Weiguang Huang "*Numerical Investigation of a Stagnation Point Reverse Flow Combustor*". ASME Conf. Proc. 2008, 507 (2008)
- [9] Undapalli Satish, Srinivasan, Srikant Menon, Suresh "*LES of premixed and non-premixed combustion in a stagnation point reverse flow combustor*", Proceedings of the Combustion Institute, Volume 32, Issue 1, 2009, Pages 1537-1544
- [10] R.M.C. So, S.A. Ahmed, H.C. Mongia, "An experimental investigation of gas jets in confined swirling air flow", NASA CR-3832 (1984)
- [11] Liang Wang, "*Experimental and Computational Investigation of Thermal-Flow Characteristics of Gas Turbine Reverse-Flow Combustor*", M S Engg. Thesis, University of New Orleans, B.S. Tongji University of China, 2003
- [12] Muthu selvan G, Muralidhara H S and Vinod kumar vyas. "Effect of atomizer constant on the performance of Simplex Atomizer", Proceedings of the 22nd National Conference on IC Engines and Combustion, NIT-Calicut, Kerala, India, 10-13 December, 2011, Paper No: 2-017
- [13] Muthu selvan G, Muralidhara H S , Vinod Kumar vyas, Dinesh kanth TP, Kumaran and Mahesh, "*Effect of Inlet Tangential Port Area on the Performance of Small-Scale Simplex atomizer*" ASME Gas turbine India Conference, Mumbai, Paper No GT India 2012 – 9520
- [14] Muthu selvan G, Muralidhara H S , Vinod Kumar vyas, Narendran, Karthy and Karthic, "*Effect of Inlet Tangential Port Area on Droplet Size Distribution and Primary Breakup Length of Small-Scale Simplex Atomizer*" Abstract Accepted by ASME GT-India, 2013
- [15] Glenn Research center, Chemical Equilibrium with Application, <http://www.grc.nasa.gov/WWW/CEA>